

EFFECT OF WINDOWS SIZES ON AIR CIRCULATION INSIDE BUILDING USING CFD SIMULATION

MAHESH R. NALAMWAR¹, DHANANJAY K. PARBAT² & D. P. SINGH³

¹ HOD & Assistant Professor, Department of Civil Engineering,
Jagdambha College of Engineering & Technology, Yavatmal, India

² Lecturer (Sel Grade), Department of Civil Engineering, Govt. Polytechnic, Nagpur, India

³ Principal and Professor, Department of Civil Engineering, KDK College of Engineering, Nagpur, India

ABSTRACT

Indoor air environment of residential building depends on the circulation of air inside the building and exhaust system. It is necessary to have uniform flow of air inside the building at working plane. Buildings are design with natural ventilation strategy to reduce the energy consumption of building. Natural ventilation depends on inside and outside building parameters like, outside wind speed, obstruction in room, shape of building, inlet and out let windows location size, orientation, bouncy force due to temperature variation in wind. Computational fluid dynamics can be used to study the air flow inside room. CFD Simulation results are validated with experimental data and found to be nearby. In present study, effect of windows size on air circulation was studied. Modeling of room was done in Autodesk Revit software and exported to CFD software. Different cases are considered for simulation with constant inlet window size and varying outlet window size. At working plane, velocity reading is recorded for every case. Velocity vector diagram is compared to understand the circulation of air inside the room. As per observation it is found that the velocity vectors at observation points are reducing with increase in outlet window size.

KEYWORDS: Window Size, Ventilation, Computational Fluid Dynamics (CFD), Air Circulation & Simulation

Received: Jun 15, 2017; **Accepted:** Jul 30, 2017; **Published:** Aug 01, 2017; **Paper Id.:** IJCSEIERDAUG20175

INTRODUCTION

Indoor air environment of residential building depends on the circulation of air inside the building and exhaust system. It is necessary to have uniform flow of air inside the building at working plane. Buildings are design with natural ventilation strategy to reduce the energy consumption of building. Natural ventilation depends on inside and outside building parameters like, outside wind speed, obstruction in room, shape of building, inlet and out let windows location size, orientation, bouncy force due to temperature variation in wind. With proper planning of building with respect to prevailing wind direction and opening sizes, Natural ventilation can be achieved. Model study of air circulation inside building was conducted by researchers to understand the flow path of air. To study the impact of urban environment on efficiency of natural ventilation technique, experiments are carried out in apartment at canyon in Athens, Greece. Trace gas technique was used to study single sided and cross ventilation¹.

Natural cross ventilation study with opening at non-symmetrical location was studied to understand the flow pattern inside the room. Experimental setup was prepared and reading of temperature and air velocity was measured. Experimental results are compared with numerical modeling of setup using computational fluid

dynamics. It is found that the numerical predictions obtained from all models are in acceptable agreement with experimental measurement². Building with multiple opening was investigated to study the air circulation. Cross and single side ventilation was studied using $k-\epsilon$ turbulence model. Velocity was observed at horizontal plane and at defined location in vertical direction. Air flow distribution around the building was studied using CFD simulation. From various cases, cross ventilation performs better in all respects³. Single house family house energy load was studied using Autodesk Green Building Studio[®]. Total 65 design scenarios are prepared in Autodesk Revit[®] software with variation in windows size, windows location and orientation of windows. It is observed that as WWR increases, energy load increases. When windows are located at mid height of wall, energy load was lowest⁴.

Numerical study of interaction of wind and building opens the new branch of engineering named Computational wind engineering. Using FLUENT 5 CFD software, wind effects on tall building was examined. The effect of flow condition around the building was also modeled. $k-\epsilon$ method was used for turbulence model and results are compared with measurements from wind tunnel study. Study results are helpful to understand the effect of development of construction of new towers in city. Because of construction of towers, wind pressure assumed on already constructed building may change. Two modeling methods are compared and discussion on results are done⁵.

Computational fluid dynamics (CFD) was used to study the ventilation system of health care room. Simulation provides understanding of efficiency, reliability and adequacy of ventilation system. It also provides important suggestions for controlling energy consumption, patients comfort and air quality in room. In case study an actual hospital room was investigated to study the efficiency of ventilation, heating and air-conditioning plant. Considering different events of the patients like coughing or second breathing, three 3D models are prepared and studied. For simulating dispersal of bacteria-carrying droplets, particle tracing and diffusion model was developed⁶. CFD was used by various civil engineering projects to solve the critical problems in civil engineering. With the help of CFD simulation, economical solutions can be achieved⁷.

METHODOLOGY

Autodesk CFD2016 software was used for simulation of windows size effect on air circulation in room. Model was prepared in Autodesk Revit software and exported to CFD2016⁸. As per metrological data of Nagpur city, maximum wind speed of 10 km/hr and minimum of 4 km/hr in summer⁹. For modeling purpose 6 km/hr wind speed is considered which will be at inlet windows¹⁰. Six cases are considered to study the effect of windows size opening. In every case outlet window size was changed gradually to observe the variation of air flow inside the room. Following data is used for model preparation

- Location – Nagpur
- Room Size – 4 x 4 m (L/B ratio = 1)
- Floor Height – 3m
- Windows Size – 1.5 x 1.2
- Lintel Level – 2.1 m
- Reading observation level – 1.2m
- Sill Level – 0.9m

- Walls - Brick wall material.
- Floor and Roof - Concrete floor and roof.
- Mesh Sizing - Meshing was done with Autosize. As per autosize, 10687 nodes and 39419 elements are automatically generated by software.
- Boundary Condition - At Inlet window wind, speed of 1.6 m/s (6 km/hr) was defined. At outlet window, zero pressure was defined.
- Solver Settings - In Solution control tab, Intelligent solution control was enabled. In Advection for turbulence air flow, Adv 5 was selected. Turbulence model of k- ϵ Model was used since the flow of air was with turbulence.

Inlet and outlet windows are placed on opposite wall at center of wall, as shown in Figure 1. Simulation was performed for Air circulation inside the room. Effect of windows size on flow of air circulation inside the room was studied.

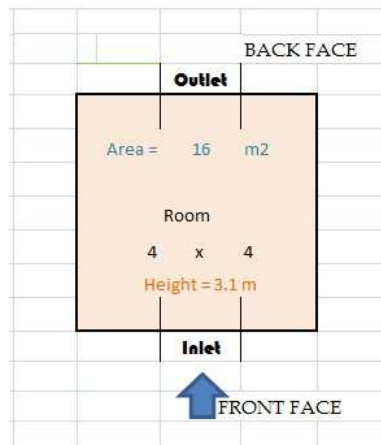


Figure 1: Room Plan Indicating Windows Location

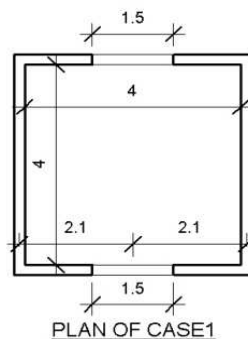


Figure 2: Plan of Case 1

Six cases are prepared to study the effect of windows size on air circulation inside room as shown in Table No.1. In all cases inlet windows size is 1.5x1.2m and outlet windows size gradually changed from 1.5 x 1.2 to 2.25 x 1.2.

Table 1: Cases Definition Table

Case	IN Window				Out Window				%	Velocity Magnitude	
No	B	H	No	Area	B	H	No	Area	Area	Mean Velocity m/s	Standard Deviation
WS1	1.5	1.2	1	1.8	1.5	1.2	1	1.8	100%	0.2810	0.3833
WS2	1.5	1.2	1	1.8	1.65	1.2	1	1.98	110%	0.2782	0.3725
WS3	1.5	1.2	1	1.8	1.8	1.2	1	2.16	120%	0.2802	0.3697
WS4	1.5	1.2	1	1.8	1.95	1.2	1	2.34	130%	0.2794	0.3627
WS5	1.5	1.2	1	1.8	2.1	1.2	1	2.52	140%	0.2801	0.3632
WS6	1.5	1.2	1	1.8	2.25	1.2	1	2.7	150%	0.2803	0.3565

SIMULATION RESULTS OUTPUT

Results are observed at working plane set at 1.2 m from floor level. Velocity magnitude was studied for all cases and velocity vector was drawn to understand direction of air.

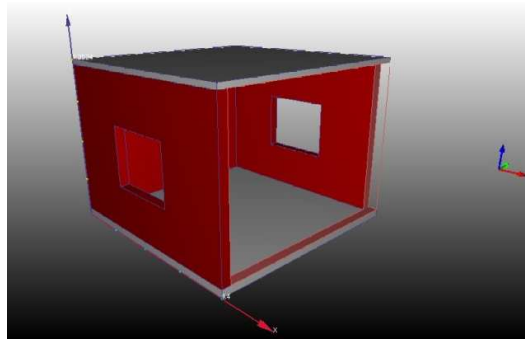


Figure 3: 3D Model in CFD Software

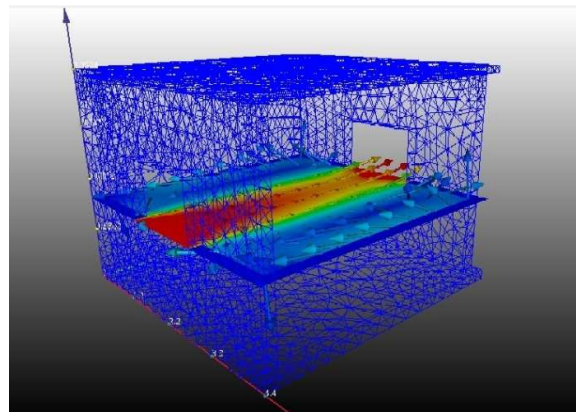


Figure 4: Velocity Magnitude Output for Case 1 In 3D

Output is in the form of Graphical Figures and reports. Following are the various graphical output for cases defined above. Left side scale shows the velocity gradient from 0 to 1.6 m/s.

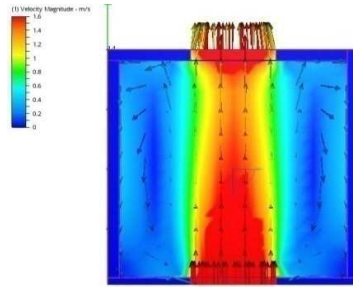


Figure 5: Velocity Magnitude Output for Case 1

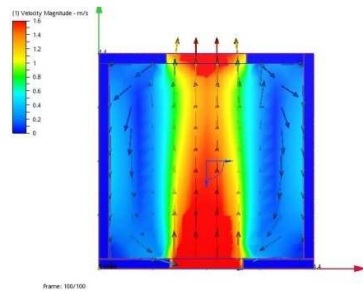


Figure 6: Velocity Magnitude Output for Case 2

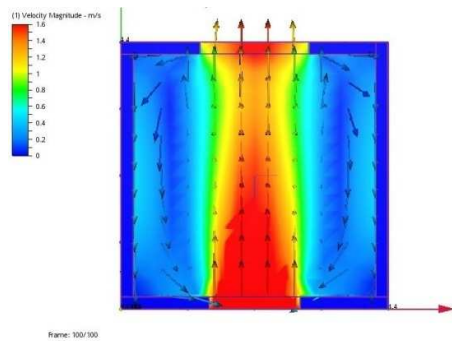


Figure 7: Velocity Magnitude Output for Case 3

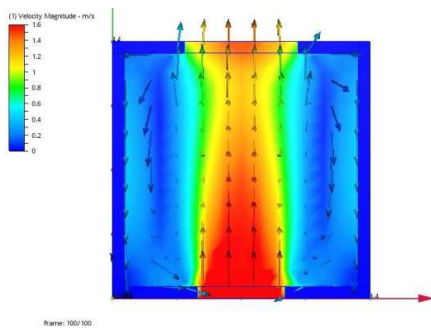


Figure 8: Velocity Magnitude Output for Case 4

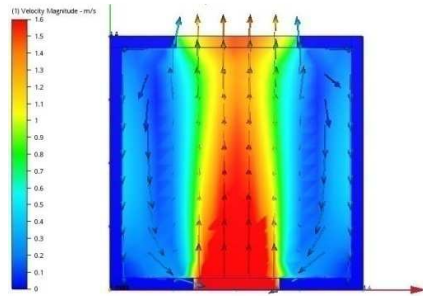


Figure 9: Velocity Magnitude Output for Case 5

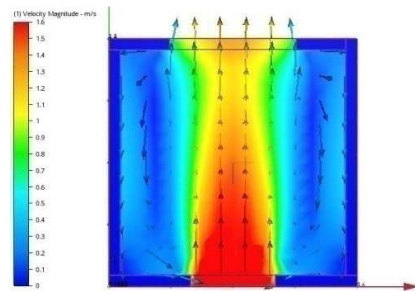


Figure 10: Velocity Magnitude Output for Case

Observation were recorded on 16 points which are spaced 1 m center to center in room as shown in drawing below.

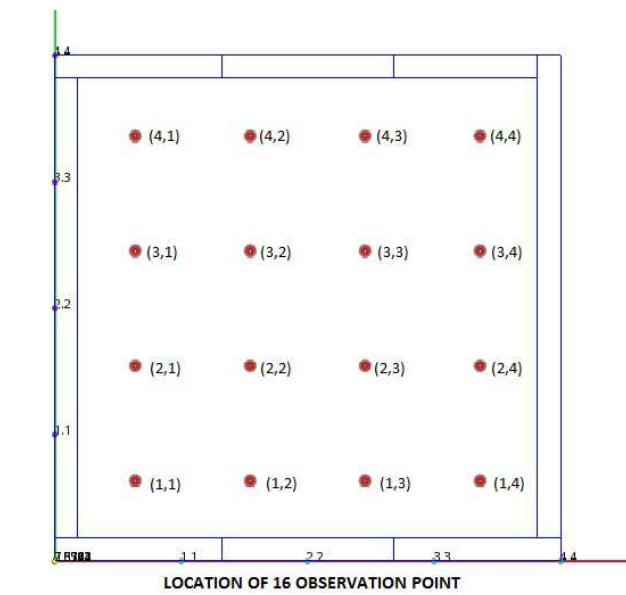


Figure 11: Observation Points Location at 1.2m Plane

Air velocity for every case was recorded on observation point for all Six cases of models. Velocity was measured in m/s.

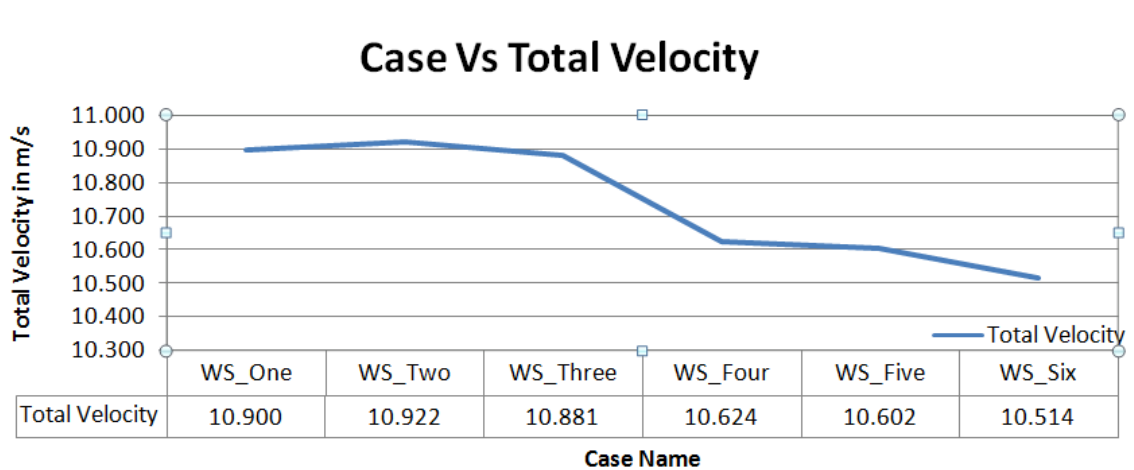
Table 2: Air Velocity at Observation Points in m/s for Various Cases

Observation Point	Velocity Magnitude in m/s					
	WS One	WS Two	WS Three	WS Four	WS Five	WS Six
P11	0.183	0.189	0.203	0.182	0.179	0.187
P12	1.520	1.522	1.519	1.514	1.515	1.518
P13	1.479	1.480	1.464	1.465	1.476	1.464
P14	0.187	0.198	0.199	0.192	0.193	0.196
P21	0.164	0.165	0.194	0.158	0.153	0.166
P22	1.256	1.248	1.253	1.235	1.231	1.239
P23	1.261	1.269	1.241	1.237	1.247	1.215
P24	0.153	0.170	0.166	0.153	0.114	0.153
P31	0.129	0.125	0.152	0.123	0.992	0.125
P32	1.054	1.032	1.051	1.018	0.929	1.025
P33	1.080	1.091	1.062	1.047	0.108	1.012
P34	0.127	0.138	0.134	0.123	0.126	0.116
P41	0.131	0.128	0.113	0.108	1.055	0.106
P42	1.007	0.978	0.992	0.955	1.009	0.934
P43	1.048	1.057	1.029	1.001	0.159	0.947
P44	0.122	0.133	0.110	0.112	0.116	0.113

SIMULATION RESULT & DISCUSSIONS

Simulation was performed to calculate the velocity at twelve observation points. In every case velocity magnitude is recorded at observation points. Figure.12 shows the graph of simulation cases and cumulative velocity at 12 observation point at working level.

Velocity at various points decreases as the size of outlet window is increased.

**Figure 12: Total Velocity on Observation Points for Various Cases**

Standard deviation of all readings at twelve points are found to reduce, as size of outlet window increases(Figure 13). Reduced standard deviation shows that air velocity is not distributed evenly on all points. Hence as we increase the outlet window size as compared to inlet window size, velocity distribution in room is not uniform.

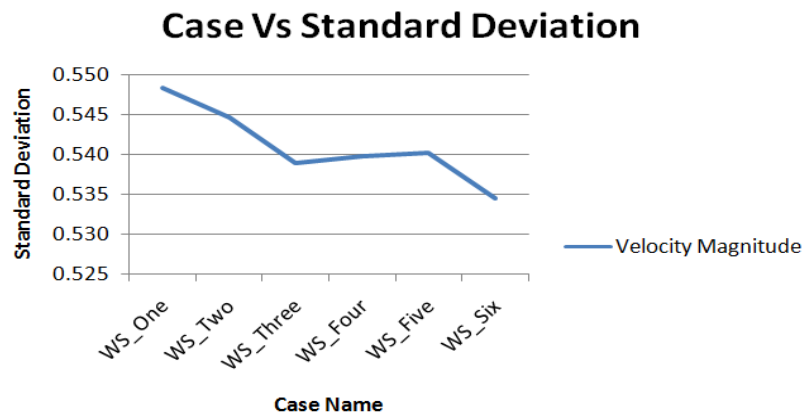


Figure 13: Deviation Velocity on Observation Points due to Various Cases

CONCLUSIONS

In CFD modeling, Outlet window size is increased from 100 to 150 %. With constant air speed of 1.6 m/s from inlet, velocity was measured at 12 observation points at elevation of 1.2m from floor level. It is observed that velocity decreases with increase in outlet window area or size as compared to inlet window. As per standard deviation comparison, air circulation is concentrated at middle of room and air distribution is not uniform. Effective air circulation also depends on obstructions inside the room and climatic condition.

REFERENCES

1. Georgakis C, Santamouris M. Experimental investigation of air flow and temperature distribution in deep urban canyons for natural ventilation purposes. 2006;38:367-376. doi:10.1016/j.enbuild.2005.07.009.
2. Adamu ZA, Price ADF. Natural Ventilation with Heat Recovery: A Biomimetic Concept. 2015:405-423. doi:10.3390/buildings5020405.
3. Bangalee MZI, Lin SY, Miao JJ. Wind driven natural ventilation through multiple windows of a building : A computational approach. Energy Build. 2012;45:317-325. doi:10.1016/j.enbuild.2011.11.025.
4. Kim S, Zadeh PA, Staub-french S, Froese T, Terim B. Assessment of the Impact of Window Size, Position and Orientation on Building Energy Load Using BIM. Procedia Eng. 2016;145:1424-1431. doi:10.1016/j.proeng.2016.04.179.
5. Swaddiwudhipong S, Khan MS. Dynamic response of wind-excited building using CFD. J Sound Vib. 2002;253(4):735-754. doi:10.1006/jsvi.2000.3508.
6. Balocco C. Hospital ventilation simulation for the study of potential exposure to contaminants. Build Simul. 2011;4(1):5-20. doi:10.1007/s12273-011-0019-6.
7. Nalamwar MR, Parbat DK, Singh DP. Application of CFD software for planning and design of civil engineering structures. Res J Eng Sci. 2017;6(3):48-51.
8. Computational Fluid Dynamics Software | Autodesk CFD. <http://www.autodesk.com/products/cfd/overview>. Accessed May 12, 2014.
9. Nandankar PK, Dewangan PL, Surpam R V. CLIMATE OF NAGPUR. 1st ed. Nagpur: Regional Metrological Centre Airport, Nagpur (M.S.); 2011.

10. ENERGY PLUS. Weather Data For Nagpur / EnergyPlus. https://energyplus.net/weather-location/asia_wmo_region_2/IND/IND_Nagpur.428670_ISHRAE. Accessed February 3, 2017.

AUTHORS BIOGRAPHY



Mr. Mahesh R. Nalamwar has done his B.E. in civil engineering from Amravati University in 1997 with Gold Medal and M.E. degrees from Shivaji University in 2001. Presently he is working as HOD Civil Engg. Department at Jagadambha College of engineering and technology, Yavatmal, India. He has 10 years of industrial experience and 7 years of teaching experience. His area of specialization is Sustainable building, Green Structural design and Computational Fluid Dynamics. He has Published 5 research papers in International Journals and attended various conferences. Is a member of Insts (IEI).



Dr. Dhananjay K. Parbat holds a B.E. (Civil Engineering) degree from Nagpur University, Post Graduation in Transportation Engineering from SGSITS, Indore (MP), PhD (Civil Engg) from SGB Amravati University. He is Lecturer in Civil Engg, Govt. Polytechnic, Nagpur, Maharashtra., having total teaching experience of 24 years. He supervises PhD scholars at RTM Nagpur and SGB Amravati University. He is Fellow of Institution of Engg (India), LM-ISTE, LM-AIFUCTO, LMICI, LMIRC. He has worked as member, Board of studies, SGB Amravati University, Amravati and working as program coordinator and subject expert in designing curriculum for MSBTE, Mumbai. He has published 82 technical research papers in reputed international, national journals and conferences. He is a recipient of Maharashtra State Best Teacher Award and ISTE Best Polytechnic Teacher Award.



Dr. D. P. Singh has done his B. E. in Civil Engineering from Nagpur University, Maharashtra, India and M. Tech in Environmental Engineering from VRCE, Maharashtra, India and PhD from Nagpur University. Presently he is working as a Principal at K. D. K. College of Engineering, Nagpur. He has published more than 42 papers in National/ International Conference and journal. He has 32 years of experience of teaching. He was Chairman of PG Civil Engineering board of RTMNU, Nagpur. He was also a member of BOS Civil Engineering of RTMNU, Nagpur. He is a life member of Institution of Engineers (IEI), Indian Concrete Institute.

